

## Version 25 Software Update Details

---

### Problem Fixes in Version 25.0.2 (13-Dec-2021)

#### Duplicate

- Duplicating components could result in pin names and/or numbers being visible on the new component when they were not shown on the original one.

#### Forward Design Changes

- The Component Bin was being displayed during Forward Design Changes even when no components had been added to the bin.

#### GenCAD Output

- The Description field on components was being incorrectly truncated to 20 characters when the GenCAD specification does not stipulate a maximum length for this field.

#### Gerber Output

- An incorrect aperture was being generated for drilled-out pads (those with the drill hole larger than the pad land).

#### New Component

- Leaving the Default Reference field blank on the New Component dialog was causing the wrong error message to be displayed.

#### Reports

- Option added to Preferences dialog to show program version in report headers.
- 

### Problem Fixes in Version 25.0.1 (11-Nov-2021)

#### Add To Net

- Using Add To Net on an unconnected pin in a Schematic could cause the application to quit.

#### Cursors

- It was not always possible to choose the 'small cross' cursor type.

#### Databook

- Resetting the working files was not removing 'thumbnail' image files.

#### Duplicate

- If two or more pins on a schematic gate were assigned to the same PCB pin, then duplicating that component in a Schematic design could cause the set of 'common' pins to be reduced to a single pin.

#### GenCAD output

- Values added to a component instance in a design were not being included in the output file.

#### Import Eagle

- Attempting to import some specific Eagle SCH files would cause the application to quit.

#### Intelligent Gerber Import

- Files could be listed on the dialog as using "mm" for units when they were in fact inches.
- The progress dialog did not show the name of the file currently being imported.
- Loading a saved settings (.IGI) file did not restore the NC Drill entries.
- Sometimes the import process would create unnecessary additional empty PCB designs.

#### Plotting

- When plotting a panel with rotated PCB instances, some pads were not being output at the correct orientation.
- When plotting a panel to Excellon (NC Drill), the panel outline was not included unless you checked 'Board', then it would also plot the PCB board outlines.
- When using the 'auto mask' feature to automatically generate solder or paste mask for copper shapes in a PCB symbol, sometimes these shapes would not appear in the output file.

#### Projects

- When working with a project file that contained references to PDF documents, launching the application with that project file open could cause the application to quit.

**Properties**

- When viewing the properties of a Pad, it was not possible to add that pad to a net if it was not already connected.

**Reports**

- The Library report of components that use symbols could omit any items that had lower-case letters in their name.

**Tool Bar**

- Double-clicking on the tool bar icon for Add Layer Stack would cause the application to quit.

Copyright © 1997-2021 WestDev Ltd

Number One Systems and Easy-PC are trademarks of WestDev Ltd

All trademarks acknowledged to their rightful owners. E&OE